



# Chapter 4

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## ***Advanced Dimensioning Techniques and Base Feature Options***

### **Learning Objectives**

**After completing this chapter, you will be able to:**

- *Dimension the sketch using the Autodimension Sketch tool.*
- *Dimension the true length of an arc.*
- *Measure Distances and View Section Properties*
- *Create solid base extruded features.*
- *Create thin base extruded features.*
- *Create solid base revolved features.*
- *Create thin base revolved features.*
- *Dynamically rotate the view to display the model from all directions.*
- *Modify the orientation of the view.*
- *Change the display modes of the solid model.*
- *Apply materials and textures to the models.*

## ADVANCED DIMENSIONING TECHNIQUES

In this chapter, you will learn about some of the advance dimensioning techniques used in dimensioning the sketches in SolidWorks. With this release of SolidWorks, you will be able to apply all possible dimensions to a sketch using a single option, which is known as **Autodimension Sketch**. The advanced dimensioning techniques are discussed next.

### Automatically Dimensioning Sketches

<b>CommandManager:</b>	Dimension/Relations > Autodimension (Customize to Add)
<b>Menu:</b>	Tools > Dimensions > Autodimension Sketch
<b>Toolbar:</b>	Dimension/Relations > Autodimension (Customize to Add)



The **Autodimension** tool is used to automatically apply the dimensions to the sketch. To apply autodimensions to a sketch, draw the sketch using standard sketching tools and then apply the required relations to the sketch. Now, choose the **Autodimension** button from the **Dimension/Relations CommandManager** to display the **Autodimension PropertyManager**, as shown in Figure 4-1. The options available in this **PropertyManager** are discussed next.

### Entities to Dimension

The **Entities to Dimension** rollout is used to specify the entities to which the dimensions have to be applied. The **All entities in sketch** radio button is selected by default. As a result, all entities drawn in the current sketching environment are selected to apply the dimensions. The **Selected entities** radio button is selected if you have to dimension only the selected entities. When you select this radio button, the **Selected Entities to Dimension** selection box will be displayed in the **Entities to Dimension** rollout. Select the entities to be dimensioned using the select cursor. The names of the selected entities will be displayed in the **Selected Entities to Dimension** selection box. If you select one or more entities before invoking the **Autodimension sketch PropertyManager**, the **Selected entities** radio button will be selected by default. Also, the names of the selected entities will be displayed in the **Selected Entities to Dimension** selection box.

### Horizontal Dimensions

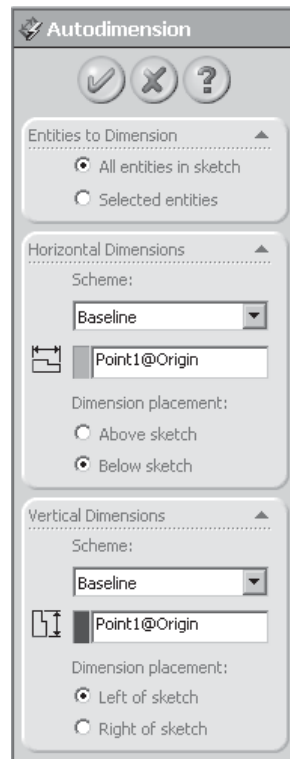
The **Horizontal Dimensions** rollout is used to specify the type of horizontal dimension, reference for the horizontal dimension, and the dimension placement. The options available in the **Horizontal Dimensions** rollout are discussed next.

#### Scheme

The **Scheme** drop-down list is used to specify the dimensioning scheme to be applied to the sketch. Various types of dimensioning schemes available in this drop-down list are discussed next.

#### Chain

The **Chain** option is used for the relative or incremental horizontal dimensioning of the sketch. When you invoke the **Autodimension PropertyManager** and select this scheme, a point or a vertical line will be selected as the reference entity. This reference entity is used as a datum for generating dimensions. The name of the selected reference



**Figure 4-1** The *Autodimension PropertyManager*

entity is displayed in the **Datums - Vertical Model Edge, Model Vertex, Vertical Line or Point** selection box and the reference entity is displayed in pink in the drawing area. You can also specify a user-defined reference entity.



#### Note

*Chain dimensioning should be avoided if the tolerances relative to a common datum are required in the part.*

#### Baseline

The **Baseline** option is used for absolute or datum vertical dimensioning of the sketch. In this dimensioning method, the dimensions are applied to the sketch with respect to the common datum. When you invoke the **Autodimension PropertyManager** and select this option, a point or a vertical line will be selected as the reference entity, which is used as a datum for generating dimensions. The name of the selected reference entity will be displayed in the **Datums - Vertical Model Edge, Model Vertex, Vertical Line or Point** selection box and the reference entity will be displayed in pink in the drawing area. You can also specify a user-defined reference entity.

#### Ordinate

The **Ordinate** option is used for the ordinate dimensioning of the sketch. When you invoke the **Autodimension PropertyManager** and select this option, a point or a vertical

line will be selected as the reference entity, which will be used as a datum for generating dimensions. The name of the selected reference entity is displayed in the **Datums - Vertical Model Edge, Model Vertex, Vertical Line or Point** selection box and the reference entity will be displayed in pink in the drawing area. You can also specify a user-defined reference entity.

#### **Dimension placement**

The **Dimension placement** area is used to define the position where the generated dimensions will be placed. Two radio buttons are available in this area. The first is the **Above sketch** radio button. If you use this option, the horizontal dimensions generated using the **Autodimension** tool will be placed above the sketch. The **Below sketch** radio button is selected by default and is used to place the dimensions below the sketch.

### **Vertical Dimensions**

The **Vertical Dimensions** rollout is used to specify the type of vertical dimension, reference for the vertical dimension, and the dimension placement. The options available in the **Scheme** area of this rollout are similar to those available in the **Horizontal Dimensions** rollout. The remaining option is discussed next.

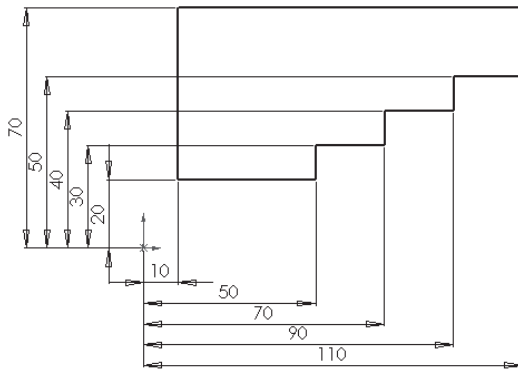
#### **Dimension placement**

The **Dimension placement** area of the **Vertical Dimensions** rollout is used to define the position where the generated dimensions will be placed. The **Left of the sketch** radio button is selected to place the dimensions on the left of the sketch. The **Right of the sketch** radio button is selected to place the dimensions on the right of the sketch. The **Left of the sketch** radio button is selected by default.

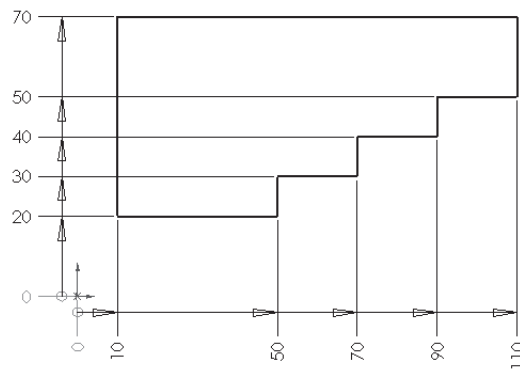
After specifying all parameters in the **Autodimension Sketch PropertyManager**, choose the **OK** button or choose the **OK** icon from the **Confirmation Corner**. The dimension, with the selected dimension scheme, will be applied to the sketch. Figure 4-2 shows the autodimension created using the **Baseline** scheme taking the origin as the datum. Figure 4-3 shows the autodimension created using the **Ordinate** scheme taking the origin as the datum.

### **Dimensioning the True Length of an Arc**

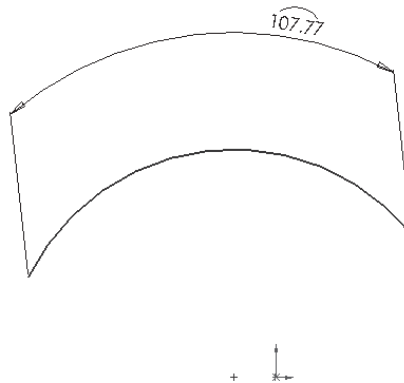
In SolidWorks, you can also apply the dimension of the true length of an arc, which is one of the advantages of the sketching environment of SolidWorks. To apply the dimension of the true length, invoke the **Smart Dimension** tool and select the arc using the dimension cursor. A radial dimension will be attached to the cursor. Move the cursor to any of the endpoints of the arc. When the cursor snaps the endpoint, use the left mouse button to specify the first endpoint of the arc. A linear dimension will be attached to the cursor; move the cursor to the second endpoint of the arc and when the cursor snaps the endpoint, select it. A dimension will be attached to the cursor. Move the cursor to an appropriate place to place the dimension. The dimension of the true length of the arc is shown in Figure 4-4.



**Figure 4-2** Baseline dimension created using the *Autodimension* tool



**Figure 4-3** Ordinate dimension created using the *Autodimension* tool



**Figure 4-4** Dimensioning the true length of an arc

## MEASURING DISTANCES AND VIEWING SECTION PROPERTIES

In SolidWorks, you can measure the distance of the entities and also view the section properties. These tools are discussed next.

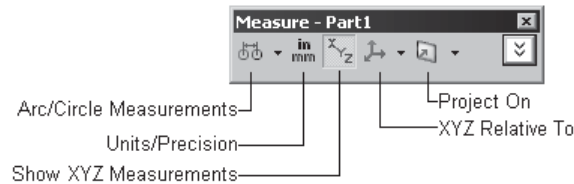
### Measuring Distances

<b>CommandManager:</b>	Tools > Measure
<b>Menu:</b>	Tools > Measure
<b>Toolbar:</b>	Tools > Measure



The **Measure** tool is used to measure the perimeter, angle, radius, and distance between lines, points, surfaces, and planes in sketches, 3D models, assemblies, or drawings. To use the measure tool, invoke the **Measure** toolbar by choosing the

**Measure** button from the **Tools CommandManager**; the **Measure** toolbar will be displayed. The name of the document in which you are working will be displayed at the top of the **Measure** toolbar, refer to Figure 4-5. The current cursor will be replaced by the measure



*Figure 4-5 The Measure toolbar*

cursor. Using the measure cursor, select the entity or entities to be measured. The result related to the selected element or elements will be displayed in the callout or callouts attached to the selected entity or entities. You can also view the result in the **Measure** toolbar by expanding it. To expand the toolbar, choose the button with down arrows on the right of the toolbar. The options in the **Measure** dialog box are discussed next.

### Arc/Circle Measurements

The **Arc/Circle Measurements** button is used to specify the technique of measuring the distance between selected arcs or circles. When you choose the **Arc/Circle Measurements** button a flyout will appear. The **Center to Center** option will be selected by default in this flyout. With this option selected, the center-to-center distance will be measured when you select two arcs or circles. If you choose the **Minimum Distance** option from this cascading menu, the minimum distance between the selected arcs or circles will be measured, which is the minimum tangential distance between the two arcs or circles. However, if you choose the **Maximum Distance** option from this cascading menu, the maximum tangential distance between the selected arcs or circles will be measured.

### Units/Precision

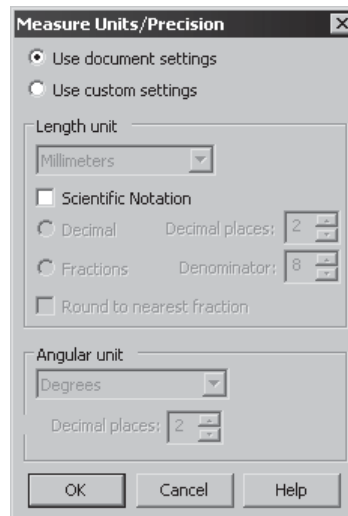
The **Units/Precision** button is used to set the type of units and their precision. To set the type and the precision of units, choose this button; the **Measure Units/Precision** dialog box will be displayed as shown in Figure 4-6. The **Use document settings** radio button is selected by default in this dialog box. The default units and precision of the document are used while measuring the entities. You can also set the type of units and their precision for measuring the entities. To do so, select the **Use custom settings** radio button from this dialog box. Other options available in this dialog box will be invoked. These options are discussed next.

#### Length unit Area

The **Length unit** area is used to set the units and options of linear measurements of the entities. The **Unit** drop-down list is provided at the top right corner of the **Length unit** area. In this drop-down list, you can select any type of unit such as **Angstroms**, **Nanometers**, **Microns**, **Millimeters**, **Centimeters**, **Meters**, **Microinches**, **Miles**, **Inches**, **Feet**, and **Feet and Inches**. The **Decimal places** spinner is provided to control the decimal places. The other options in the **Length unit** area are discussed next.

#### Decimal

The **Decimal** radio button is available only when you select the units as **Microinches**, **Miles**, **Inches**, or **Feet and Inches** from the **Unit** drop-down list. This radio button is selected to display the dimension in the decimal form. You can also specify the decimal



*Figure 4-6 The Measure Units/Precision dialog box*

places using the **Decimal places** spinner provided on the right of the **Decimal** radio button.

#### Fractions

The **Fractions** radio button is available only when you select the units as **Microinches**, **Miles**, **Inches**, or **Feet and Inches** from the **Units** drop-down list. This radio button is selected to display the dimension in the fraction form. You can also set the value of the denominator using the **Denominator** spin box provided on the right of the **Fractions** radio button.

#### Round to nearest fraction

The **Round to nearest fraction** check box is selected to display the value in fractions by rounding the value to the nearest fraction.

#### Scientific Notation

The **Scientific Notation** check box is selected to display the value in the scientific notation units.

#### Angular unit Area

The **Angular unit** area is used to set the units for angular measurement. This area is provided with a drop-down list to specify the angular measurement units such as **Degrees**, **Deg/Min**, **Deg/Min/Sec**, and **Radians**. The **Decimal places** spinner is provided to specify the decimal places.

### Show XYZ Measurement

The **Show XYZ measurement** button is chosen by default in the **Measure** toolbar and therefore shows the dx, dy, and dz, values of the selected entities. If you clear this button, only the minimum distance between the selected entities will be displayed.

### XYZ Relative To

The **XYZ Relative To** button is used to define the coordinate system along which the selected entities will be measured. By default, the part origin is selected as the coordinate system. To select any other coordinate system that you had created earlier, choose the **XYZ Relative To** button from the **Measure** toolbar. A flyout and all the coordinate systems that you have created along with the part origin will be displayed. Choose the coordinate system from this flyout. You will learn more about creating additional coordinate systems in the later chapters.

### Projected On

The **Projected On** button is used to specify the location where the selected entity should be projected. You can project the selected entity on the screen or on a specific plane. The system will then calculate the measurement of the true projection. To specify the location, choose the **Projected On** button from the **Measure** toolbar; a callout will be displayed and you can select the location where you need to project the selected entity.

## Determining Section Properties of Closed Sketches

<b>CommandManager:</b>	Tools > Section Properties
<b>Menu:</b>	Tools > Section Properties
<b>Toolbar:</b>	Tools > Section Properties

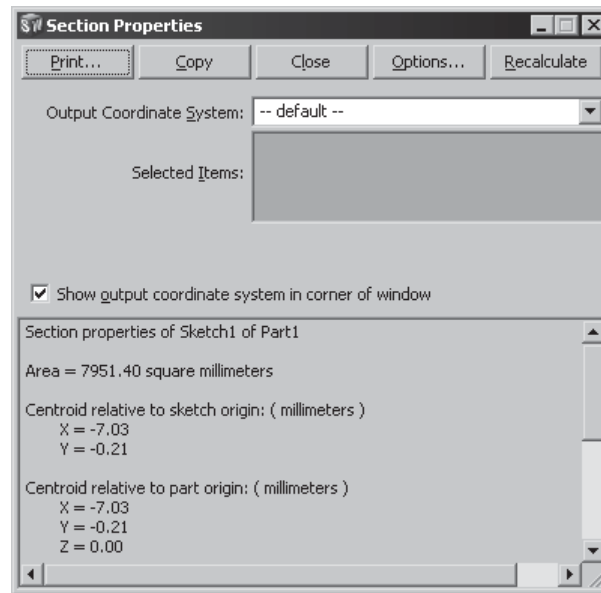


The **Section Properties** tool enables you to determine the section properties of the sketch in the sketching environment or of the selected planar face in the **Part** mode and the **Assembly** mode. Remember that the section properties of only the closed sketches with nonintersecting closed loops can be determined. The section properties include the area, centroid relative to the sketch origin, centroid relative to the part origin, moment of inertia, polar moment of inertia, angle between the principle axes and sketch axes, and principle moment of inertia.

To calculate the section properties, draw the sketch and then choose the **Section Properties** button from the **Tools CommandManager**; the **Section Properties** dialog box will be displayed as shown in Figure 4-7.

When you invoke the **Section Properties** dialog box, a 3D triad will be placed at the centroid of the sketch. The section properties of the sketch are displayed in the **Section Properties** dialog box. The **Selected Items** display box is used to display the name of the selected planar face or sketch whose section properties are to be calculated. When you are in the **Part** mode, select the face to calculate the section properties and choose the **Recalculate** button to display the properties. If you want to calculate the section properties of some other face, clear the previously selected face from the selection set. Now, select the new face and choose the **Recalculate** button.





**Figure 4-7** Resized view of the **Section Properties** dialog box

The **Print** button available in the **Section Properties** dialog box is used to print the section properties. The **Copy** button is chosen to copy the section properties to the Clip board from where you can copy them to a program such as MS Word. The rest of the options in this dialog box are similar to those discussed in the previous section.

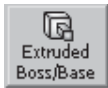
## CREATING BASE FEATURES BY EXTRUDING SKETCHES

The sketches that you have drawn until now can be converted into base features by extruding using the **Extruded Boss/Base** tool. This tool is available in the **Features CommandManager**. After drawing the sketch, choose the **Features** button from the **CommandManager** to display the **Features CommandManager**. From the **Features CommandManager**, choose the **Extruded Boss/Base** button. The sketching environment will be closed and the part modeling environment will be invoked. Also, the preview of the feature that will be created using the default options will be displayed in trimetric view. The trimetric view gives a better display of the solid feature.

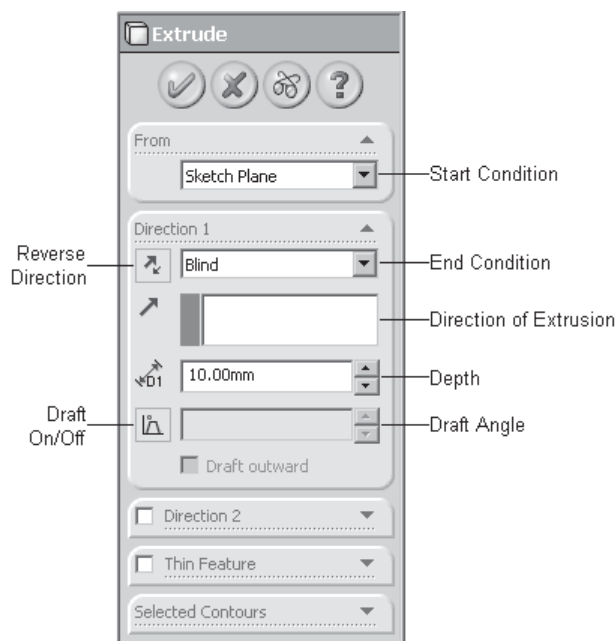
On the basis of the options and the sketch selected for extruding, the resulting feature can be a solid feature or a thin feature. If the sketch is closed, it can be converted into a solid feature or a thin feature. However, if the sketch is open, it can be converted only into a thin feature. The solid and thin features are discussed next.

## Creating Solid Extruded Features

**CommandManager:** Features > Extruded Boss/Base  
**Menu:** Insert > Boss/Base > Extrude  
**Toolbar:** Features > Extruded Boss/Base



After you have completed drawing and dimensioning the closed sketch and converted it into a fully defined sketch, invoke the **Features CommandManager** by choosing the **Features** button and then choose the **Extruded Boss/Base** button from the **Features CommandManager**. You will notice that the view is automatically changed to trimetric view and the **Extrude PropertyManager** is displayed, as shown in Figure 4-8.

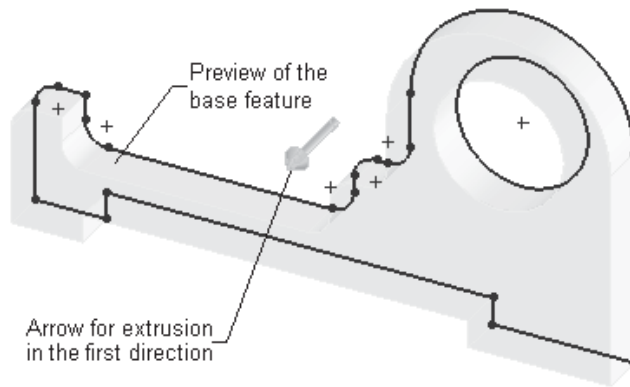


**Figure 4-8** The *Extrude PropertyManager*

You will notice that the preview of the base feature will be displayed in temporary graphics and an arrow will appear on the sketch. The arrow will appear in front of the sketch and is transparent. Figure 4-9 shows the preview of the sketch being extruded. Note that if the sketch consists of some closed loops inside the outer loop, they will be automatically subtracted from the outer loop while extruding. The options available in the **Extrude PropertyManager** are discussed next.

### Direction 1

The **Direction 1** rollout is used to specify the end condition for extruding the sketch in one direction from the sketch plane. The options available in the **Direction 1** drop-down list are discussed next.



**Figure 4-9** Preview of the feature being extruded



**Tip.** You can also extrude an underdefined or an overdefined sketch. However, if you extrude an underdefined sketch, a - sign will be displayed on the left of the sketch in the **Feature Manager**. Similarly, if you extrude an overdefined sketch, you will find a + sign on the left of the sketch in the **Feature Manager**. To check these signs, click on the + sign available on the left of **Extrude 1** in the **Feature Manager**. The sketch will be displayed and you can see the + or the - sign.

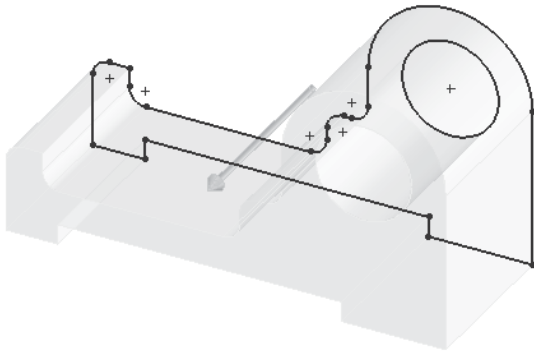
### End Condition

The **End Condition** drop-down list provides the options to define the termination of the extruded feature. Note that because this is the first feature, some of the options available in this drop-down list will not be used at this stage. Also, some additional options will be available later in this drop-down list. The options available to define the termination of the base feature are discussed next.

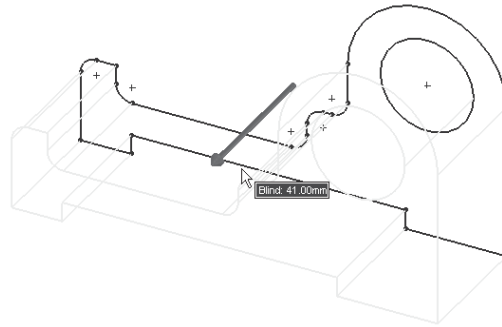
#### Blind

The **Blind** option is selected by default and is used to define the termination of the extruded base feature by specifying the depth of extrusion. The depth of extrusion can be specified in the **Depth** spinner, which will be displayed in this rollout when you select the **Blind** option. You can reverse the extrusion direction by selecting the **Reverse Direction** button provided on the left of this drop-down list. Figure 4-10 shows the preview of the feature being created by extruding the sketch using the **Blind** option.

You can also extrude a sketch to a blind depth by dynamically dragging the feature using the mouse. Invoke the **Extrude PropertyManager** and move the mouse to the transparent arrow and when the color of the arrow changes to red, press the left mouse button. Now, move the cursor to specify the depth of extrusion; the **Blind** callout will also be displayed below the cursor. The value of the depth of extrusion will change dynamically in this callout as you move the cursor. Left-click to specify the termination of the extruded feature. The preview of the sketch being dragged is shown in Figure 4-11. The select cursor will be replaced by the mouse cursor. Use the right mouse button to



**Figure 4-10** Preview of the feature being extruded using the **Blind** option



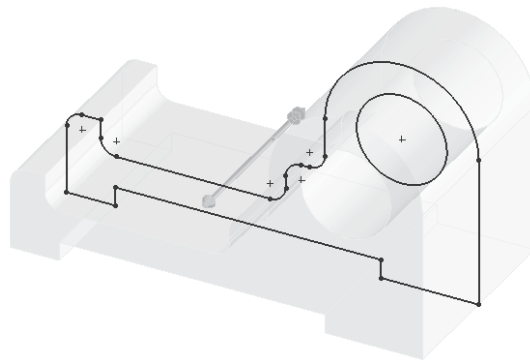
**Figure 4-11** Preview of the feature being extruded by dynamically dragging

complete the feature creation or choose the **OK** button from the **Extrude PropertyManager**.

#### **Mid Plane**

The **Mid Plane** option is used to create the base feature by extruding the sketch equally in both directions of the plane on which the sketch is drawn. For example, if the total depth of the extruded feature is 30 mm, it will be extruded 15 mm toward the front of the plane and 15 mm toward the back. The depth of the feature can be defined in the **Depth** spinner, which is displayed below this drop-down list.

Figure 4-12 shows the preview of the feature being created by extruding the sketch using the **Mid Plane** option.



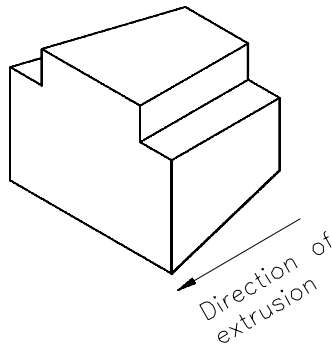
**Figure 4-12** Preview of the feature being extruded using the **Mid Plane** option

#### **Draft On/Off**

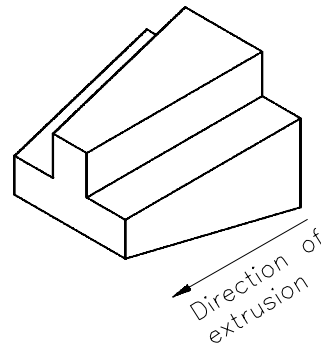
The **Draft On/Off** button is used to specify a draft angle while extruding the sketch. Applying the draft angle will taper the resulting feature. This button is not chosen by default. Therefore,

the resulting base feature will not have any taper. However, if you want to add a draft angle to the feature, choose this button; the **Draft Angle** spinner and the **Draft outward** check box will be available. You can enter the draft angle for the feature in the **Draft Angle** spinner. By default, the feature will be tapered inward, as shown in Figure 4-13.

If you want to taper the feature outward, select the **Draft outward** check box, which is displayed below the **Draft Angle** spinner. The feature created with the outward taper is shown in Figure 4-14.



**Figure 4-13** Feature created with outward draft



**Figure 4-14** Feature created with inward draft



#### Note

The **Direction of Extrusion** area will be discussed in later chapters.



**Tip.** You can also select the termination options using the shortcut menu that is displayed when you right-click in the drawing area.

## Direction 2

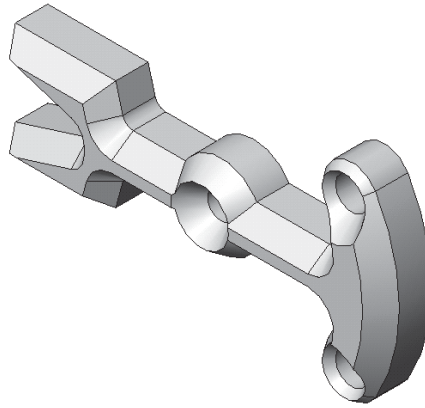
The **Direction 2** check box is selected to invoke the **Direction 2** rollout. This rollout is used to extrude the sketch with different values in the second direction of the sketching plane. This check box will not be available if you select the **Mid Plane** termination type.



**Tip.** As soon as you select the **Direction 2** check box, another arrow will be displayed in the preview of the feature. You can use this arrow to define the depth of extrusion in the second direction.

The options available in this rollout are similar to those in the **Direction 1** rollout. Note that unlike the **Mid Plane** termination option, the depth of extrusion and other parameters in both directions can be different. For example, you can extrude the sketch to a blind depth of 10 mm and an inward draft of 35-degree in front of the sketching plane and to a blind depth of 15 mm and an outward draft of 0-degree behind the sketching plane, as shown in Figure 4-15.

After setting the values for both directions, choose the **OK** button or choose the **OK** icon from the confirmation corner. The feature will be created with the defined values.



*Figure 4-15 Feature created in two directions with different values*

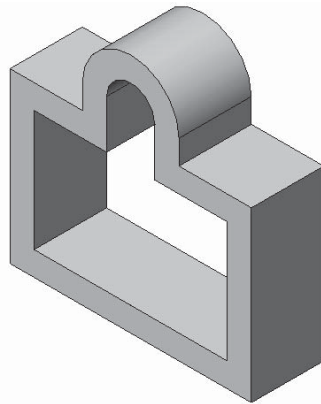


**Note**

The *Selection Contours* rollout will be discussed in later chapters.

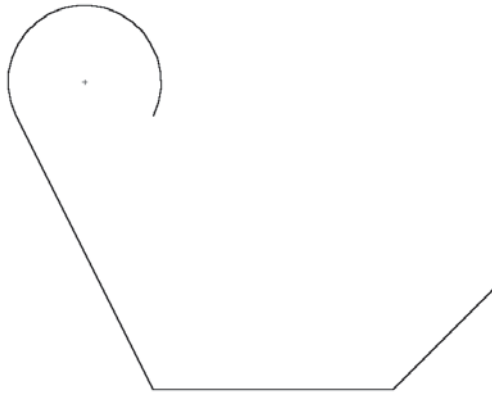
## Creating Thin Extruded Features

Thin extruded features can be created using a closed or an open sketch. If the sketch is closed, it will be offset inside or outside to create a cavity inside the feature, as shown in Figure 4-16.



*Figure 4-16 Thin feature created using a closed loop*

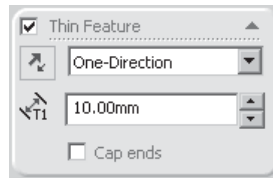
If the sketch is open, as shown in Figure 4-17, the resulting feature will be as shown in Figure 4-18. To convert a closed sketch into a thin feature, choose the **Thin Feature** check box to invoke the **Thin Feature** rollout. The **Thin Feature** rollout, as shown in Figure 4-19, is used to create a thin feature. However, if the sketch to be extruded is open, the **Thin Feature** rollout will be invoked automatically when you invoke the **Extrude PropertyManager**.



**Figure 4-17** Open loop to be converted into thin feature



**Figure 4-18** Resultant thin feature created with fillets at sharp corners



**Figure 4-19** The *Thin Feature* rollout

The options available in the **Thin Feature** rollout of the **Extrude PropertyManager** are discussed next.

## Type

The options provided in the **Type** drop-down list are used to select the method of defining the thickness of the thin feature. These options are discussed next.

### One-Direction

The **One-Direction** option is used to add the thickness on one side of the sketch. The thickness can be specified in the **Thickness** spinner provided below this drop-down list. For the closed sketches, the direction can be inside or outside the sketch. Similarly, for open sketches, the direction can be below or above the sketch. You can reverse the direction of thickness using the **Reverse** check button available on the left of this drop-down list. This button will be available only when you select the **One-Direction** option from this drop-down list.

### Mid-Plane

The **Mid-Plane** option is used to add the thickness equally on both sides of the sketch. The value of the thickness of the thin feature can be specified in the **Thickness** spinner provided below this drop-down list.

### Two-Direction

The **Two-Direction** option is used to create a thin feature by adding different thicknesses on

both sides of the sketch. The thickness values in direction 1 and direction 2 can be specified in the **Direction 1 Thickness** spinner and the **Direction 2 Thickness** spinner, respectively. These spinners will be automatically displayed below the **Type** drop-down list when you select the **Two-Direction** option from this drop-down list.

### Cap ends

The **Cap ends** check box will be displayed only when you select a closed sketch to convert into a thin feature. This check box is selected to cap the two ends of the thin extruded feature. Both ends will be capped with a face of the thickness you specify. When you select this check box, the **Cap Thickness** spinner will be displayed below this check box. The thickness of the end caps can be specified using this spinner.

### Auto-fillet corners

The **Auto-fillet corners** check box will be displayed only when you select an open sketch to convert into a thin feature. If you select this check box, all sharp vertices in the sketch will be automatically filleted while converting into a thin feature. As a result, the thin feature will have filleted edges. The radius of the fillet can be specified in the **Fillet Radius** spinner, which will be displayed below the **Auto-fillet corners** check box when you select this check box.

Figure 4-20 shows the thin feature created by extruding an open sketch in both directions. Note that a draft angle is applied to the feature while extruding in the front direction and the **Auto-fillet corners** option is selected while creating this thin feature.



**Figure 4-20** Thin feature created in both directions



#### Note

*Only the corners of the thin features that can accommodate the given radius will be filleted; other corners that cannot accommodate the given radius will not be filleted.*



## CREATING BASE FEATURES BY REVOLVING SKETCHES

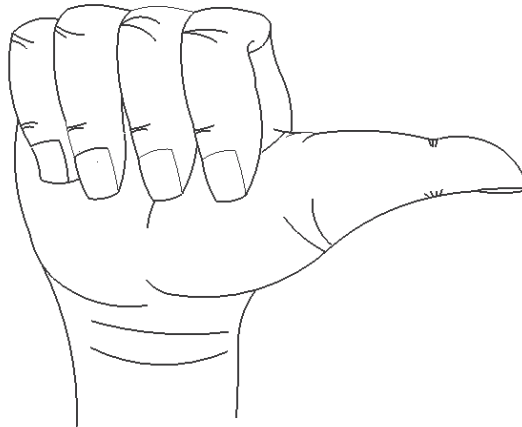
**CommandManager:** Features > Revolved Boss/Base  
**Menu:** Insert > Boss/Base > Revolve  
**Toolbar:** Features > Revolved Boss/Base



The sketches that you have drawn until now can also be converted into base features by revolving using the **Revolved Boss/Base** tool. This tool is available in the **Features CommandManager**. Using this tool, the sketch is revolved about the revolution axis.

The revolution axis could be an axis, an entity of the sketch, or an edge of another feature to create the revolved feature. Note that whether you use a centerline or an edge to revolve the sketch, the sketch should be drawn on one side of the centerline or the edge.

In SolidWorks, the right-hand thumb rule is used while determining the direction of revolution. The right-hand thumb rule states that if the thumb of your right hand points in the direction of the axis of revolution, the direction of the curled fingers will determine the default direction of revolution, see Figure 4-21.



**Figure 4-21** Right-hand thumb rule

For example, consider a case in which you draw a centerline from left to right (direction of thumb in Figure 4-21) in the drawing area. Now, if you use this centerline to create a revolved feature, the default direction of revolution will be in the direction of the curled fingers.



**Note**

*You can also reverse the direction of revolution using the button available in the **Revolve PropertyManager**.*

After drawing the sketch, as you choose this tool, you will notice that the sketching environment is closed and the part modeling environment is invoked. Similar to extruding the sketches, the resulting feature can be a solid feature or a thin feature, depending on the sketch and the options selected to be revolved. If the sketch is closed, it can be converted into a solid feature or

a thin feature. However, if the sketch is open, it can be converted only into a thin feature. The solid and thin features are discussed next.

## Creating Solid Revolved Features

After you have completed drawing and dimensioning the closed sketch and converted it into a fully defined sketch, choose the **Revolved Boss/Base** button from the **Features** toolbar. You will notice that the view is automatically changed to a 3D view, and the **Revolve PropertyManager** is displayed, as shown in Figure 4-22, as well as the confirmation corner. Also, the preview of the base feature, as will be created using the default options, will be displayed in temporary shaded graphics. The direction arrow will also be displayed in gray. If you have not drawn the centerline, you will be prompted to select an axis of revolution. Select an edge or entity you need to define as an axis of revolution. The options available in the **Revolve Parameters** rollout of the **Revolve PropertyManager** are discussed next.

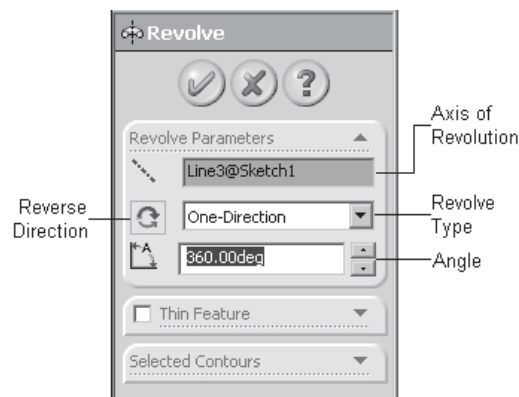


Figure 4-22 The Revolve PropertyManager



**Tip.** Even if you can revolve the sketch using an edge in the sketch, you need to draw a centerline. This is because you can create linear diameter dimensions for the revolved features only using a centerline.

## Revolve Type

The **Revolve Type** drop-down list provides the options to define the termination of the revolved feature. The options available in this drop-down list to terminate the revolved feature are discussed next.

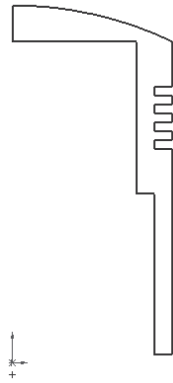
### One-Direction

The **One-Direction** option is used to revolve the sketch on one side of the plane on which it is drawn. The angle of revolution can be specified in the **Angle** spinner displayed below

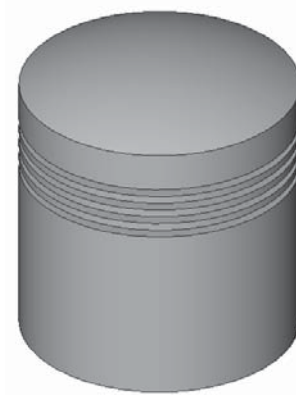


**Tip.** You can dynamically specify the angle in a revolved feature by dragging the direction arrows. You can also use the right mouse button to display the shortcut menu; the options available in the **PropertyManager** are also available in the shortcut menu.

this drop-down list. The default value of the **Angle** spinner is **360deg**. Therefore, if you revolve the sketch using this value, a complete round feature will be created. You can also reverse the direction of revolution of the sketch by choosing the **Reverse Direction** button, which will be displayed when you select this option. Figure 4-23 shows the sketch of the piston and Figure 4-24 shows the resulting piston created by revolving the sketch through an angle of 360-degree. Note that the outer left vertical edge of the sketch that is vertically in line with the origin is used to revolve the sketch.

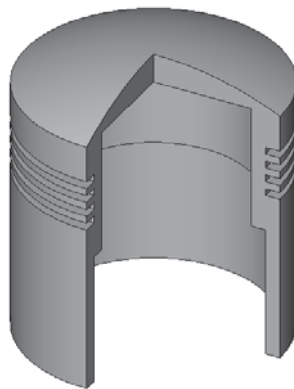


**Figure 4-23** Sketch of the piston to be revolved



**Figure 4-24** Feature created by revolving the sketch through an angle of 360-degree

Figure 4-25 shows a piston created by revolving the same sketch through an angle of 270-degree.



**Figure 4-25** Feature created by revolving the sketch through an angle of 270-degree

### Mid-Plane

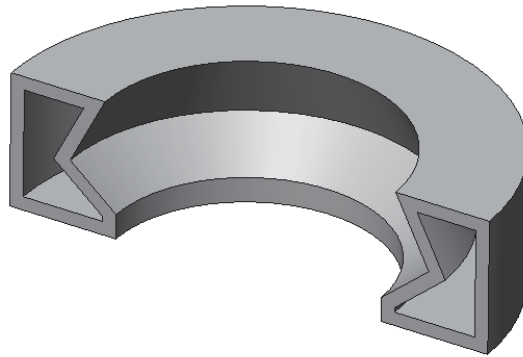
The **Mid-Plane** option is used to revolve the sketch equally on both sides of the plane on which it is drawn. The angle of revolution can be specified in the **Angle** spinner. When you choose this option, the **Reverse Direction** button will be unavailable.

**Two-Direction**

The **Two-Direction** option is used to create a revolved feature by revolving the sketch using different values on both sides of the plane on which it is drawn. The angle values in direction 1 and direction 2 can be specified in the **Direction 1 Angle** spinner and the **Direction 2 Angle** spinner, respectively. These spinners will be displayed below the **Revolve Type** drop-down list automatically when you select the **Two-Direction** option from this drop-down list.

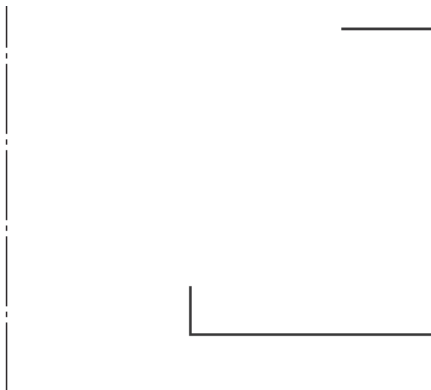
**Creating Thin Revolved Features**

Thin revolved features can be created using closed or open sketches. If the sketch is closed, it will be offset inside or outside to create a cavity inside the feature as shown in Figure 4-26. In this figure, the sketch is revolved through an angle of 180-degree.

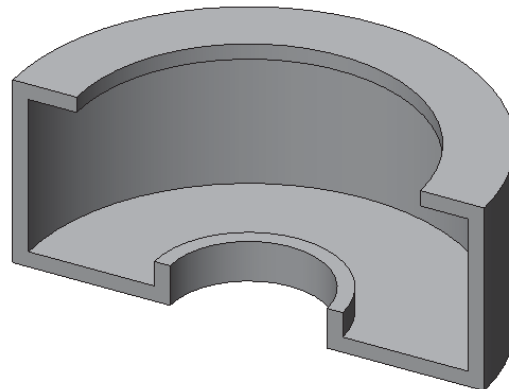


**Figure 4-26** Thin feature created by revolving the sketch through an angle of 180-degree

If the sketch is open, as shown in Figure 4-27, the resulting feature will be similar to that shown in Figure 4-28.



**Figure 4-27** The open sketch to be revolved and the centerline to revolve the sketch



**Figure 4-28** Thin feature created by revolving the open sketch through an angle of 180-degree

To convert a closed sketch into a thin feature, select the **Thin Feature** check box from the **Revolve PropertyManager** to invoke the **Thin Feature** rollout. However, if the sketch to be revolved is open and you invoke the **Revolved Boss/Base** tool, the **SolidWorks** information box will be displayed. This information box will inform you that the sketch is currently open and a non-thin revolved feature requires a closed sketch. You will be given an option of automatically closing the sketch. If you choose **Yes** from this dialog box, a line segment will be automatically drawn between the first and the last segment of the sketch and the **Revolve PropertyManager** will be displayed. However, if you choose **No** from this information box, the **Revolve PropertyManager** will be displayed and the **Thin Feature** rollout will be displayed automatically. The options available in the **Thin Feature** rollout of the **Revolve PropertyManager**, shown in Figure 4-29, are discussed next.



Figure 4-29 The **Thin Feature** rollout

## Type

The options provided in the **Type** drop-down list are used to select the method of defining the thickness of the thin feature. These options are discussed next.

### One-Direction

The **One-Direction** option is used to add the thickness on one side of the sketch. The thickness can be specified in the **Direction 1 Thickness** spinner provided below this drop-down list. For the closed sketches, the direction can be inside or outside the sketch. Similarly, for open sketches, the direction can be below or above the sketch. You can reverse the direction of thickness using the **Reverse Direction** button available on the right of this drop-down list. This button will be available only when you select the **One-Direction** option from this drop-down list.

### Mid-Plane

The **Mid-Plane** option is used to add the thickness equally on both sides of the sketch. The value of the thickness of the thin feature can be specified in the **Direction 1 Thickness** spinner provided below this drop-down list.

### Two-Direction

The **Two-Direction** option is used to create a thin feature by adding different thicknesses on both sides of the sketch. The thickness values in direction 1 and direction 2 can be specified in the **Direction 1 Thickness** spinner and the **Direction 2 Thickness** spinner, respectively.



**Tip.** While defining the wall thickness of a thin revolved feature, remember that the wall thickness should be added such that the centerline does not intersect with the sketch. If the centerline intersects with the sketch, the sketch will not be revolved.

These spinners will be displayed below the **Type** drop-down list automatically when you select the **Two-Direction** option from this drop-down list.

## DYNAMICALLY ROTATING THE VIEW OF THE MODEL

In SolidWorks, you can dynamically rotate the view in the 3D space so that the solid models in the current document can be viewed from all directions. This allows you to visually maneuver around the model so that all its features can be clearly viewed. This tool can be invoked even when you are inside some other tool. For example, you can invoke this tool when the **Extrude Feature** dialog box is displayed. You can freely rotate the model in the 3D space or rotate it around a selected vertex, edge, or face. Both methods of rotating the model are discussed next.

### Rotating the View Freely in 3D Space

<b>Toolbar:</b>	View > Rotate View
<b>Menu:</b>	View > Modify > Rotate



To rotate the view freely in the 3D space, choose the **Rotate View** button from the **View** toolbar. You can also invoke this tool by choosing the **Rotate View** option from the shortcut menu that will be displayed when you right-click in the drawing area. When you are inside some other tool, use the right mouse button in the drawing area and choose the **Zoom/Pan/Rotate > Rotate View** option from the shortcut menu to invoke the rotate view tool. When you invoke this tool, the cursor will be replaced by the rotate view cursor. Now, press the left mouse button and drag the cursor to rotate the view. Figure 4-30 shows a model being viewed from different directions by rotating the view.



*Figure 4-30 Rotating the view to display the model from different directions*

### Rotating the View Around a Selected Vertex, Edge, or Face

To rotate the view around a selected vertex, edge, or face, invoke this tool and move the rotate view cursor close to the vertex, edge, or the face around which you want to rotate the view. When it is highlighted, select it using the left mouse button. Next, drag the cursor to rotate the view around the selected vertex, edge, or face.



**Tip.** To resume rotating the view freely after you have completed rotating it around a selected vertex, edge, or face, double-click anywhere in the drawing area. Now when you drag the cursor, you will notice that the view is rotated freely in the 3D space.

If a three-button mouse is configured to your computer, you can press and drag the middle mouse button to rotate the model freely in the 3D space. Note that in this case the rotate view cursor will not be displayed.

## MODIFYING THE VIEW ORIENTATION

As mentioned earlier, when you invoke the **Extruded Boss/Base** tool or the **Revolved Boss/Base** tool, the view will be automatically changed to a trimetric view and the preview of the model will be displayed. In SolidWorks, you can manually change the view orientation using some predefined standard views or user-defined views. To invoke these standard views, choose the **Standard Views** button from the **View** toolbar; a flyout will be displayed, as shown in Figure 4-31. You can select the required standard view from this flyout.

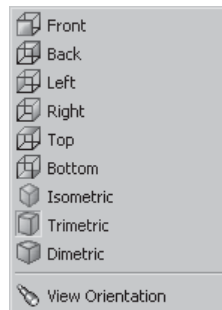


Figure 4-31 The Standard View flyout

You can also orient the view of the model by clicking on the pop-up menu button provided on the lower left corner of the drawing area. As soon as you click on this button, a pop-up menu is displayed, as shown in Figure 4-32. Apart from the standard views, this pop-up menu also provides the **Normal To** option. This option is chosen to reorient the view normal to a selected face or plane. To do this, select the face normal to which you need to reorient the model and choose the **Normal To** option from this flyout. If you have not selected a face before invoking this option, the **Normal To PropertyManager** will be displayed and you will be prompted to select a reference along which the view will be reoriented.

The other options provided in this pop-up menu are discussed later in this chapter.

You can also invoke these standard views using the **Orientation** dialog

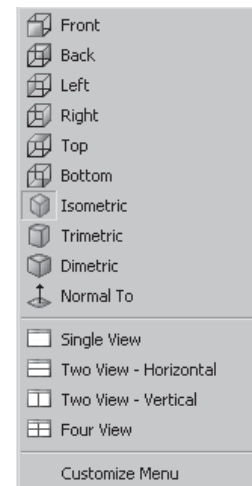
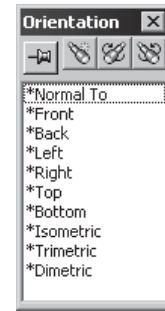


Figure 4-32 The pop-up menu

box. This dialog box is invoked by choosing the **View Orientation** option from the shortcut menu. This dialog box can also be invoked by pressing the SPACEBAR on the keyboard. Note that when you invoke this dialog box by pressing the SPACEBAR, the dialog box will be displayed at the location where the cursor is placed currently. The **Orientation** dialog box is shown in Figure 4-33.

You can invoke the view from this dialog box by double-clicking on it. You will notice that in addition to the standard views, two more views are displayed. The buttons available on top of this dialog box are discussed next.



**Figure 4-33**  
*The Orientation dialog box*

### Push-Pin



You will notice that the **Orientation** dialog box is automatically closed when you select a view or a point somewhere on the screen, or invoke a tool. If you want this dialog box to be retained on the screen, you can pin it at a location by choosing the **Push-Pin** button. This is the first button on top of this dialog box. Move the dialog box to the desired location and then choose this button. The dialog box will be pinned to that location and will not close when you perform any operation.

### New View



The **New View** button is chosen to create a user-defined view and save it in the list of views in the **Orientation** dialog box. Using various drawing display tools and the **Rotate View** tool, modify the current view and then choose this button. When you choose this button, the **Named View** dialog box will be displayed. Enter the name of the view in the **View Name** edit box and then choose the **OK** button. You will notice that a user-defined view is created and it is saved in the list available in the **Orientation** dialog box.

### Update Standard Views



The **Update Standard Views** button is chosen to modify the orientation of the standard views. For example, if you want that the view that is displayed when you invoke the **Back** option from this dialog box should be the front view, change the current view to the back view by double-clicking on it in the **Orientation** dialog box. Now, select the **Front** option from the list of views available in the **Orientation** dialog box and then choose the **Update Standard Views** button. The **SolidWorks** warning box will be displayed and you will be informed that if you change the standard view, all other named views in the model will also be changed. If you make the change using the **Yes** button, the current view that was originally the back view will become the front view. Also, all other views will be modified automatically.

### Reset Standard Views



The **Reset Standard Views** button is chosen to reset the standard settings of all standard views in the current drawing. When you choose this button, the **SolidWorks** warning box will be displayed and you will be prompted to confirm whether you want to reset all standard views to their original settings or not. If you choose **Yes**, all standard views will be reset to their default settings.



## RESTORING PREVIOUS VIEW



In SolidWorks, while working on a model or while viewing it from different directions, you need to temporarily change the view of the model. Once you have finished editing or viewing the model in that view, you can restore the previous view using the **Previous View** button in the **View** toolbar. This tool saves ten previous views of the model.

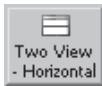
## DISPLAYING THE DRAWING AREA IN VIEWPORTS

With this release of SolidWorks, you can display the drawing area in multiple viewports. The procedure to do so is discussed next.

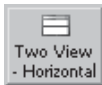


### Displaying the Drawing Area in Two Horizontal Viewports

<b>CommandManager:</b>	Standard Views > Two View - Horizontal (Customize to Add)
<b>Menu:</b>	Window > Viewport > Two View - Horizontal
<b>Toolbar:</b>	Standard Views > Two View - Horizontal (Customize to Add)



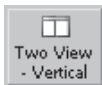
The **Two View - Horizontal** option is used to split the drawing view to display the model in two viewports placed horizontally, as shown in Figure 4-34. To display the model in this fashion, choose **Window > Viewport > Two View - Horizontal** from the menu bar. You will notice, that the drawing area is divided in two rows placed horizontally. The model is placed in the top orientation in the upper row and in the front orientation in the lower row. The type of orientation of both the models is displayed in the pop-up button provided on the lower left corner of each viewport.



To switch back to the single viewport, choose the **Single View** button from the **Standard Views** toolbar or choose **Window > Viewport > Single View** from the menu bar.

### Displaying the Drawing Area in Two Vertical Viewports

<b>CommandManager:</b>	Standard Views > Two View - Vertical (Customize to Add)
<b>Menu:</b>	View > Viewport > Two View - Vertical
<b>Toolbar:</b>	Standard Views > Two View - Vertical (Customize to Add)



The **Two View - Vertical** button is used to split the drawing view to display the model in two viewports placed vertically as shown in Figure 4-35. To display the model in this fashion, choose **Window > Viewport > Two View - Vertical** from the menu bar. You will notice, that the drawing area is divided in two columns placed vertically. The model is placed in the front orientation in the left column and in the right orientation in the right column.

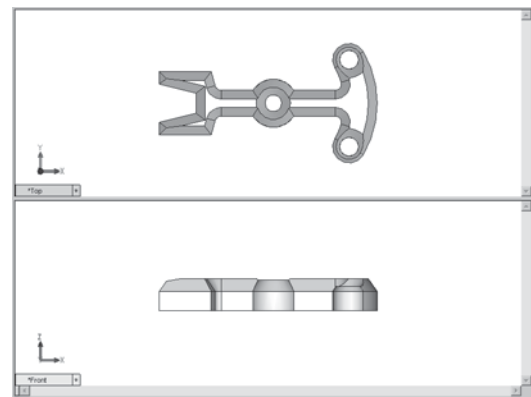


Figure 4-34 Drawing area divided in two rows horizontally using the **Two View - Horizontal** tool

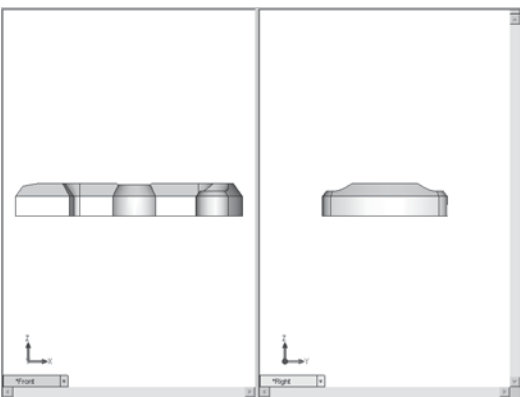


Figure 4-35 Drawing area divided in two columns vertically using the **Two View - Vertical** tool

Displaying the Drawing Area in Four Viewports

CommandManager:	Standard Views > Four Views	(Customize to Add)
Menu:	View > Viewport > Four Views	
Toolbar:	Standard Views > Four Views	(Customize to Add)



The **Four View** option is used to split the drawing view to display the model in four viewports, as shown in Figure 4-36. To display the model in this fashion, choose **Window > Viewport > Four** from the menu bar. You will notice, that the drawing area is divided in four parts. The model is placed in the front, top, right, and trimetric orientations in each viewport. The types of orientation of all models are displayed in the pop-up button provided on the lower left corner of each viewport.

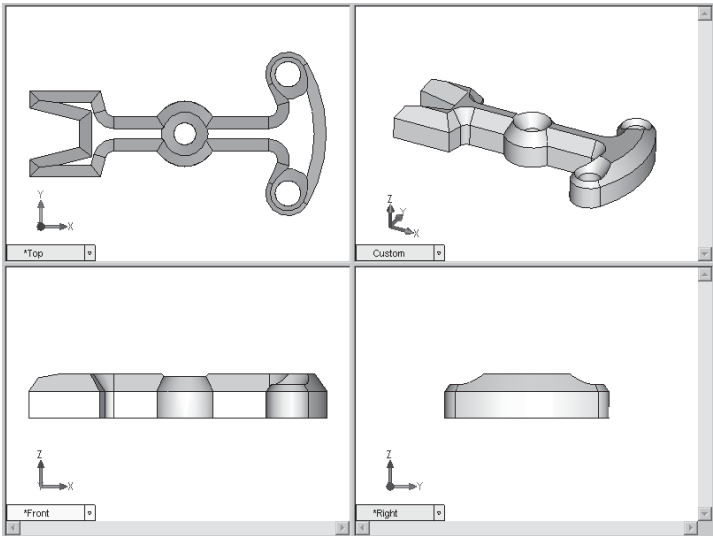


Figure 4-36 Drawing area divided in four parts using the **Four View**



**Tip.** On creating multiple viewports, you will notice that one of the pop-up buttons is highlighted, which implies that the viewport is currently active. To activate another viewport, click once in the drawing area in that viewport. After activating a viewport, you can change the model display in it. You will learn more about changing the model display later in this chapter.

You can choose **Window > Viewport > Link Views** from the menu bar to link the viewports. Now, if you pan or zoom one view while multiple viewports are being displayed, all the model in all the other viewports will pan or zoom accordingly. Note, that this linking is not applicable for the viewport in which the model is displayed in 3D orientation. To break this link, choose **Window > Viewport > Link Views** from the menu bar.

## DISPLAY MODES OF THE MODEL

SolidWorks provides you with various predefined modes to display the model. You can select any of these display modes from the **View** toolbar. These modes are discussed next.

### Wireframe



When you choose the **Wireframe** button, all hidden lines will be displayed along with the visible lines in the model. Sometimes it becomes difficult to recognize the visible lines and the hidden lines if you set this display mode for complex models.

### Hidden Lines Visible



When you choose the **Hidden Lines Visible** button, the model will be displayed in the wireframe and the hidden lines in the model will be displayed as dashed lines.

### Hidden Lines Removed



When you choose the **Hidden Lines Removed** button, the hidden lines in the model will not be displayed. Only the edges of the faces visible in the current view of the model will be displayed.

### Shaded With Edges



The **Shaded With Edges** mode is the default mode in which the model is displayed. In this display mode, the model is shaded and the edges of the visible faces of the model are displayed.

### Shaded



This display mode is similar to the **Shaded With Edges** mode, with the only difference that the edges of the visible faces will not be displayed.

### Shadows In Shaded Mode



The **Shadows In Shaded Mode** button is used to display the shadow of the model. A light appears from the top of the model to display the shadow in the current view. With this option activated, the performance of the system is affected during the dynamic



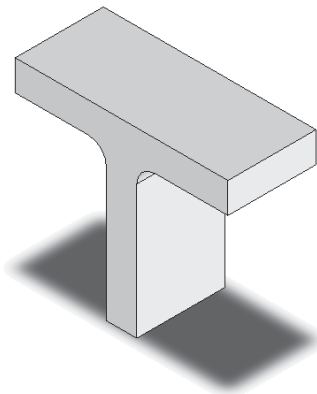
**Tip.** Sometimes when you rotate the view of a large assembly or a model with large number of features with **Shaded** or the **Hidden Lines Removed** shading modes, the regeneration of the model takes a lot of time. This can be avoided by choosing the **Draft Quality HLR/HLV** button in combination with the other shading modes. This button is not available by default, therefore, you need to customize the toolbar or **CommandManager**. Choosing this button speeds up the regeneration time and you can easily rotate the view. This is a toggle mode and is turned on when you choose this button. This button can be chosen in combination with any of the other display modes.

orientation. Remember that the position of the shadow is not changed when you rotate the model in the 3D space. To change the placement of the shadow, first remove the shadow in the shaded model using the **Shadow in Shaded Mode** button and rotate the model. After rotating the model, use the same button to activate the shadow in the shaded mode. Figure 4-37 shows a T-section with shadow in the shaded mode.

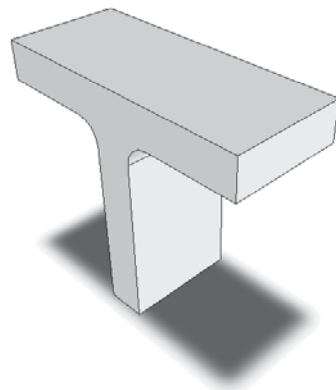
## Perspective



The **Perspective** button is not available in the **View** toolbar by default. You need to customize this toolbar to add this button. Using this button, you can display the perspective view of a model. Figure 4-38 shows the T-section with shadow in perspective view.



**Figure 4-37** Shadow in active shaded mode



**Figure 4-38** T-section displayed in perspective view with shadow



**Tip.** You can also save a perspective view as a named view. To do this, invoke the perspective view and define the orientation by rotating the view. Now, press the SPACEBAR to invoke the **Orientation** dialog box. In this dialog box, choose the **New View** button and specify the name of the view in the **Named View** dialog box. The named view will be saved in the **Orientation** dialog box and you can invoke it whenever required.

You can also modify the settings of the perspective view. Choose **View > Modify > Perspective**

from the menu bar to invoke the **Perspective View PropertyManager**. Using the **Object Sizes Away** spinner of this **PropertyManager**, you can modify the observer's position.

## ASSIGNING MATERIALS AND TEXTURES TO THE MODELS

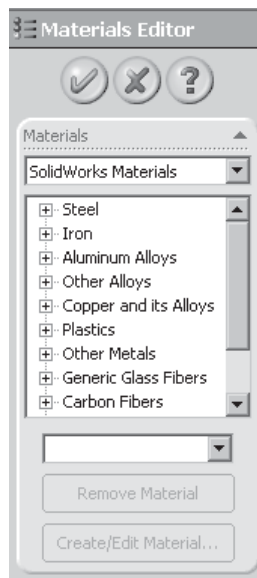
You can assign materials and textures to the models. When you apply a material to a model, the physical properties such as density, young's modulus, and so on will be assigned to the model. When you apply a texture to a model or its face, the image of that texture will be applied to the model or its selected face. Physical properties are not applied to the model when you apply the texture. The method of assigning materials and textures is discussed next.

### Assigning Materials to the Model

**Toolbar:** Standard > Edit Material (Customize to Add)  
**Menu:** Edit > Appearance > Material



Whenever you assign a material to a model, all physical properties of the selected material are also assigned to the model. As a result, when you calculate the mass properties of the model, they will be based on the physical properties of the material. To assign a material to a model, choose the **Edit Material** button from the **Standard** toolbar; the **Materials Editor PropertyManager** will be displayed, as shown in Figure 4-39.



*Figure 4-39 Partial display of the Materials Editor PropertyManager*

A number of material families are available in the **Materials** rollout. Click on the + sign located on the left of the material family to display all materials under that family. Select the material from that family.

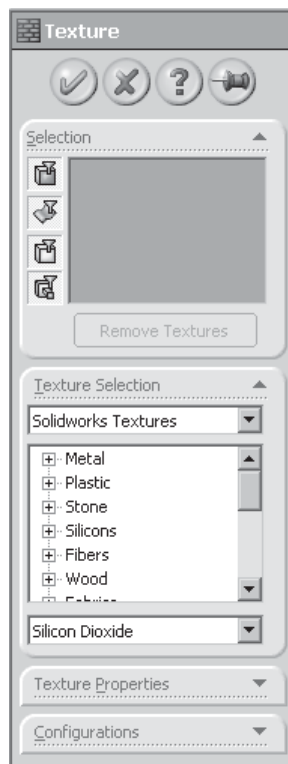
You can change the scale and angle of the textures in the selected material using the **Scale** and **Angle** spinners available in the **Visual Properties** rollout.

## Assigning Textures to the Model

**Toolbar:** Standard > Edit Texture  
**Menu:** Edit > Appearance > Texture



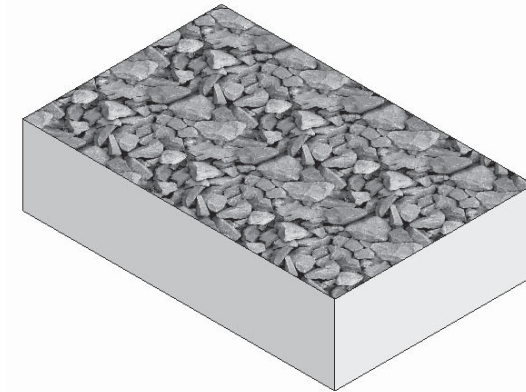
In SolidWorks 2006, you can assign a texture to a model, feature, or a selected face. This can be done using the **Texture PropertyManager**, which will be displayed when you invoke the **Edit Texture** tool. Figure 4-40 shows this **PropertyManager**.



*Figure 4-40 Partial display of the **Texture PropertyManager***

The **Selection** rollout has some buttons available on the left. These buttons are the filters for making a selection to assign the textures. For example, if you want to assign the texture on a face of the model, choose the **Select Faces** button and clear all remaining buttons. This allows you to select only a specified face. You can select the required face from the model. The selected face or model is displayed in the **Selected Entities** area of the **Selection** rollout.

Various texture families that you can select are available in the **Texture Tree** area of the **Texture Selection** rollout. Click on the + sign located on the left of a texture family to select a texture of that family. Figure 4-41 shows a model with the Gravel type of stone applied to its top face.



*Figure 4-41 Texture applied to the top face of a model*

The preview of the selected texture is displayed on the model or the face and also in the **Texture Preview** area of the **Texture Properties** rollout. You can use the **Scale** and **Angle** spinners available in this rollout to change the scale and angle of the selected texture.

You can remove the texture applied to a face or a model by choosing the **Remove Textures** button available below the **Selected Entities** area of the **Selection** rollout.

The drop-down list available below the **Texture Tree** area of the **Texture Selection** rollout lists the last ten textures used. You can also select a texture from this drop-down list.

## TUTORIALS

### Tutorial 1

In this tutorial, you will open the sketch drawn in Tutorial 2 of Chapter 3. You will then convert this sketch into an extruded model by extruding it in two directions as shown in Figure 4-42. The parameters for extruding the sketch are given next.

#### Direction 1

**Depth = 10 mm**

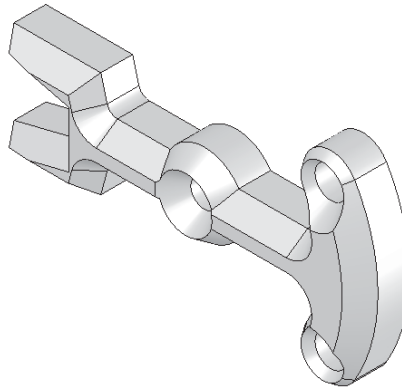
**Draft angle = 35-degree**

#### Direction 2

**Depth = 15 mm**

**Draft angle = 0-degree**

After creating the model, you will rotate the view using the **Rotate View** tool and then modify the standard views such that the front view of the model becomes the top view. You will then save the model with the current settings. **(Expected time: 30 min)**



**Figure 4-42** Model for Tutorial 1

The steps that will be followed to complete this tutorial are given next.

- a. Open the document of Tutorial 2 of Chapter 3, refer to Figure 4-43.
- b. Save this document in the *c04* folder with a new name.
- c. Invoke the **Extruded Boss/Base** tool and convert the sketch into a model, refer to Figures 4-44 and 4-45.
- d. Rotate the view using the **Rotate View** tool to view the model from all directions, refer to Figure 4-46.
- e. Invoke the **Orientation** dialog box and then modify the standard view, refer to Figure 4-47.

### Opening the Document of Tutorial 2 of Chapter 3

As the required document is saved in the *\My Documents\SolidWorks\c03* folder, you need to select this folder and then open the *c03tut2.sldprt* document.

1. Start SolidWorks 2006 by double-clicking on its shortcut icon on the desktop of your computer.
2. Choose the **Open a Document** option from the **SolidWorks Resources Task Pane** to display the **Open** dialog box.
3. Browse and select the *\My Documents\SolidWorks\c03* folder. All documents that were created in Chapter 3 are displayed in this folder.
4. Select the *c03tut2.sldprt* document and then choose the **Open** button. Close the **SolidWorks Resources Task Pane**.

Because the sketch was saved in the sketching environment in Chapter 3, it is opened in the sketching environment.

### Saving the Document in the c04 Folder

It is recommended that when you open a document of some other chapter, you should save it in the folder of the current chapter with some other name before proceeding with modifying



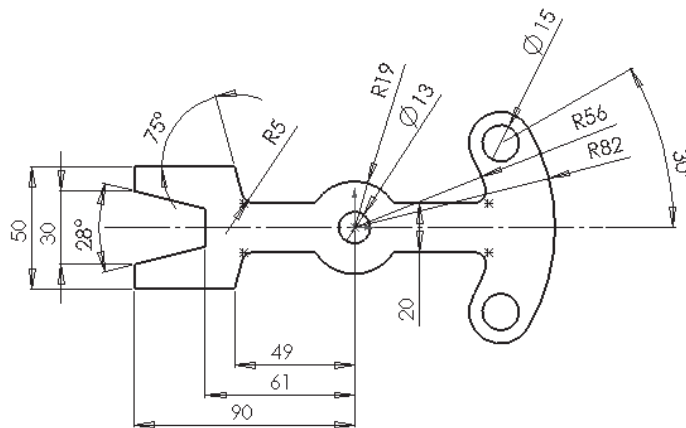
the document. This is because if you save the document in the folder of the current chapter, the original document of the other chapter will not be modified.

1. Choose **File > Save As** from the menu bar to display the **Save As** dialog box.

Because the *c03* folder was selected last to open the document, it will be the current folder.

2. Choose the **Up One Level** button available on the right of the **Save in** drop-down list to move to the *\SolidWorks* folder. Create a new folder with the name *c04* using the **Create New Folder** button. Make the *c04* folder current by double-clicking on it.
3. Enter the new name of the document as *c04tut1* in the **File name** edit box and then choose the **Save** button to save the document.

The document is saved with the new name and the new document is now opened in the drawing area, as shown in Figure 4-39.



**Figure 4-43** Sketch that will be opened in the drawing area

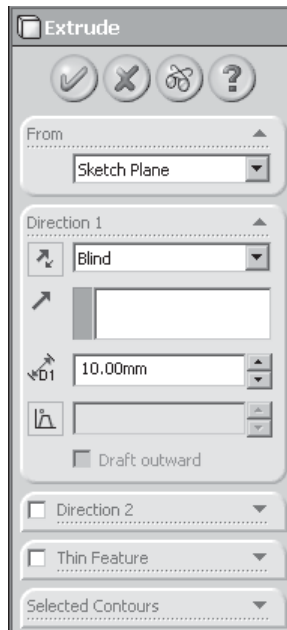
## Extruding the Sketch

Next, you need to invoke the **Extruded Boss/Base** tool and extrude the sketch using the parameters given in the tutorial description.

1. Choose the **Features** button from the **CommandManager** to display the **Features CommandManager**. From this **CommandManager**, choose the **Extruded Boss/Base** button. The sketch is automatically oriented in the trimetric view and the **Extrude PropertyManager** is displayed, as shown in Figure 4-44.



Because the sketch to be converted into a feature is closed, only the **Direction 1** rollout is displayed in the **Extrude PropertyManager**. The preview of the feature in the temporary shaded graphics with the default values is shown in the drawing area.



*Figure 4-44 The Extrude PropertyManager*

2. Choose the **Draft On/Off** button from the **Direction 1** rollout and then set the value of the **Draft Angle** spinner to **35**.

These are the settings in direction 1. Next, you need to specify the settings for direction 2.

3. Select the **Direction 2** check box to invoke the **Direction 2** rollout.

You will notice that the default values in this rollout are the same as you specified in the **Direction 1** rollout. Because the **Draft On/Off** button is chosen when you invoke the **Direction 2** rollout, you need to turn this button off. This is because you do not require the draft angle in the second direction.


4. Choose the **Draft On/Off** button from the **Direction 2** rollout to turn this option off. Set the value of the **Depth** spinner to **15** as the depth in the second direction is 15 mm. This completes all settings for the model in both directions.
5. Choose the **OK** button to create the feature or choose **OK** from the confirmation corner.

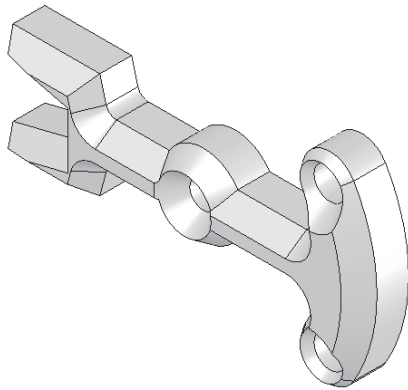
It is recommended that you change the view to isometric view after creating the feature so that you can view the feature properly.

6. Choose the **Standard Views** button from the **View** toolbar to invoke the flyout and then choose **Isometric** from this flyout. Turn off the display of the origin in the model by choosing **View > Origin** from the menu bar. The isometric view of the resulting solid model is shown in Figure 4-45.

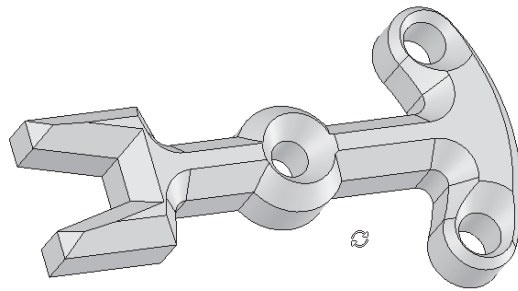
### Rotating the View

As mentioned earlier, you can rotate the view so that you can view the model from all directions.

1. Choose the **Rotate View** button from the **View** toolbar; the arrow cursor is replaced by the rotate view cursor. 
2. Press and hold the left mouse button down and drag the cursor in the drawing area to rotate the view, as shown in Figure 4-46.



**Figure 4-45** Isometric view of the solid model



**Figure 4-46** Rotating the view to display the model from different directions

You will notice that the model is being displayed from different directions. Remember that when you rotate the view, the model is not being rotated. The camera that is used to view the model is being rotated around the model.

3. After viewing the model from all directions, choose **Isometric** from the flyout that is displayed when you choose the **Standard Views** button from the **View** toolbar.

### Modifying the Standard Views

As mentioned in the tutorial description, you need to modify the standard views such that the front view of the model becomes the top view. This is done using the **Orientation** dialog box.

1. Press the SPACEBAR on the keyboard to invoke the **Orientation** dialog box.

The orientation dialog box is automatically closed as soon as you perform any other operation. Therefore, you need to pin this dialog box so that it is not closed automatically.

2. Hold the **Orientation** dialog box by selecting it on the blue bar on the top of this dialog box and then drag it to the top right corner of the drawing area.

3. Choose the **Push Pin** button to pin this dialog box at the top right corner of the drawing area. Pinning the dialog box ensures that the dialog box is not automatically closed when you perform any other operation.



4. Double-click on the **Front** option in the list box of the **Orientation** dialog box.

The current view is automatically changed to the front view and the model is now reoriented and displayed from the front.

5. Select the **Top** option from the list box by selecting it once.

Make sure you do not double-click on this option. This is because if you double-click on this option, the model is reoriented and displayed from the top.

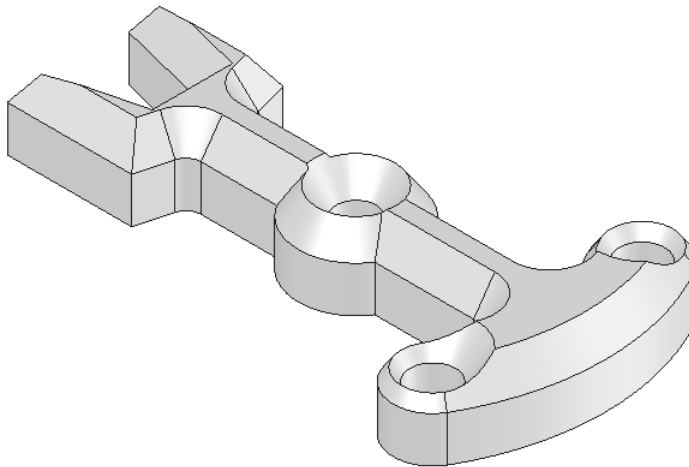
6. Now, choose the **Update Standard Views** button to update the standard views.



The **SolidWorks** warning box is displayed and you are warned that modifying the standard views will change the orientation of any named view in this document.

7. Choose **Yes** from this warning box to modify the standard views.

8. Now, double-click on the **Isometric** option provided in the list box of the **Orientation** dialog box. You will notice that the isometric view is different now, see Figure 4-47.



*Figure 4-47 Model displayed from modified isometric view*

9. Choose the **Push Pin** button in the **Orientation** dialog box again and select a point anywhere in the drawing area to close the dialog box.



## Saving the Model

As the name of the document was specified at the beginning, you just have to choose the save button now to save the document.

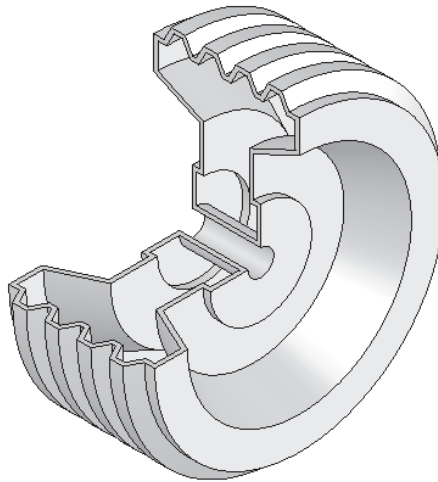
1. Choose the **Save** button from the **Standard** toolbar to save the document.

The model is saved with the name `\My Documents\SolidWorks\c04\c04tut1.sldprt`.

2. Choose **File > Close** from the menu bar to close the document.

## Tutorial 2

In this tutorial, you will open the sketch drawn in Exercise 1 of Chapter 3. You will then create a thin feature by revolving the sketch through an angle of 270-degree, as shown in Figure 4-48. You will offset the sketch outward while creating the thin feature.



*Figure 4-48 Revolved model for Tutorial 2*

After creating the model, you will turn the option on to display the shadows and also apply Copper material to the model. **(Expected time: 30 min)**

The steps that will be followed to complete this tutorial are given next.

- a. Open the sketch of Exercise 1 of Chapter 3, refer to Figure 4-49.
- b. Save it in the folder of the current chapter.
- c. Invoke the **Revolved Boss/Base** tool and revolve the sketch through an angle of 270-degree, refer to Figure 4-51.
- d. Change the current view to isometric view and then display the model in shadow, refer to Figure 4-52.
- e. Assign copper material to the model, refer to Figure 4-53.

## Opening the Document of Exercise 1 of Chapter 3

As the required document is saved in the `\My Documents\SolidWorks\c03` folder, you need to select this folder and then open the `c03exr1.sldprt` document.

1. Choose the **Open** button from the **Standard** toolbar to display the **Open** dialog box. The *c04* folder is current in this dialog box.
2. Browse and select the *\My Documents\SolidWorks\c03* folder. All documents that were created in Chapter 3 are displayed in this folder.
3. Select the *c03exr1.sldprt* document and then choose the **Open** button. The document is opened in the sketching environment.

### Saving the Document in the c04 Folder

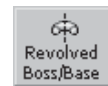
As mentioned earlier, it is recommended that you save the document with a new name in the folder of the current chapter so that the original document is not modified.

1. Choose **File > Save As** from the menu bar to display the **Save As** dialog box. Because the *c03* folder was selected last to open the document, it is the current folder.
2. Browse and select the *c04* folder and double-click on it to make it current.
3. Enter the name of the document in the **File name** edit box as *c04tut2*. Choose the **Save** button to save the document. The sketch that is displayed in the drawing area is shown in Figure 4-49.

### Revolving the Sketch

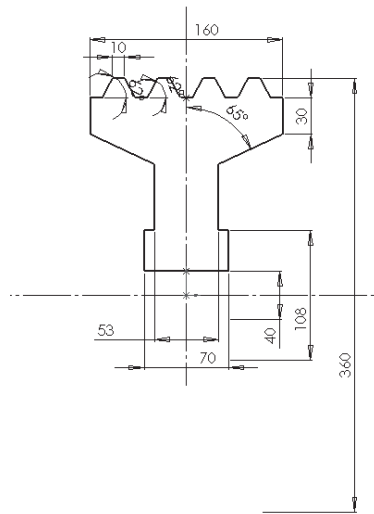
The sketch consists of two centerlines. The first centerline was used to mirror the sketched entities and the second was drawn to apply linear diameter dimensions. You need to revolve the sketch around the second centerline.

1. Choose the **Features** button from the **CommandManager** to display the **Features CommandManager**. From this **CommandManager**, choose the **Revolved Boss/Base** button. The sketch is automatically oriented in the trimetric view and the **Revolve PropertyManager** is displayed.



Because the sketch has two centerlines, SolidWorks cannot determine which one to use as an axis of revolution. This is the reason why you are prompted to select the axis of revolution.

2. Select the horizontal centerline that was used to create linear diameter dimensions as the axis of revolution. The preview of a complete revolved feature in temporary shaded graphics is displayed in the drawing area. Because the preview of the model is not displayed properly in the current view, you need to zoom the drawing.
3. Choose the **Zoom to Fit** button from the **View** toolbar or press the F key on the keyboard.
4. Set the value of the **Angle** spinner in the **Revolve PropertyManager** to **270**; the preview of the revolved model is also modified accordingly. If you enter the value in the **Angle** spinner, you need to click anywhere on the screen to make sure the preview is modified.



**Figure 4-49** Sketch for the revolved model

Note that if the horizontal centerline was drawn from left to right, then the direction of revolution has to be reversed to get the required model; refer to the right-hand thumb rule. You can reverse the direction of revolution using the **Reverse Direction** button available on the left of the **Revolve Type** drop-down list.

5. Select the **Thin Feature** check box to invoke the **Thin Feature** rollout, as shown in Figure 4-50. Set the value of the **Direction 1 Thickness** spinner to 5. You will notice that the preview of the thin feature is shown outside the original sketch.



**Figure 4-50** The **Thin Feature** rollout


6. Choose the **OK** button to create the revolved feature. You will notice that the revolved feature is created.
7. Choose the **Isometric** option from the flyout that is displayed on choosing the **Standard Views** button from the **View** toolbar.

You will notice that the sketch that was used to create the thin feature is still displayed. You need to turn off the display of this sketch.

8. Click on the + sign located on the left of **Revolve-Thin1** in the **Feature Manager Design Tree**. The tree view expands and the sketch is displayed. Right-click on the sketch and choose **Hide** from the shortcut menu to hide the sketch. The revolved feature is shown in Figure 4-51.


### Rotating the View

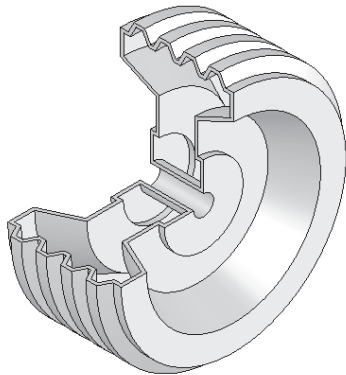
Next, you need to rotate the view so that you can view the model from all directions. As mentioned earlier, the view can be rotated using the **Rotate View** tool.

1. Choose the **Rotate View** button from the **View** toolbar; the arrow cursor is replaced by the rotate view cursor. 
2. Press the left mouse button and drag the cursor in the drawing area to rotate the view.
3. Press the SPACEBAR to invoke the **Orientation** dialog box. In this dialog box, double-click on the **Isometric** option.

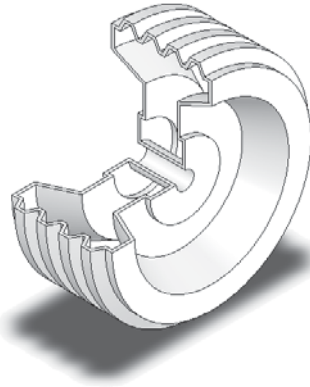
### Displaying the Shadow

As mentioned in the tutorial description, you need to display the shadow of the model if it is not displayed by default. You can turn the display of the shadow on using the **View** toolbar.

1. Choose the **Shadows In Shaded Mode** button from the **View** toolbar; the shadow of the model is displayed below the model, see Figure 4-52. 



**Figure 4-51** Model created by revolving the sketch



**Figure 4-52** Model with the display of shadow turned on

### Assigning Materials to a Model

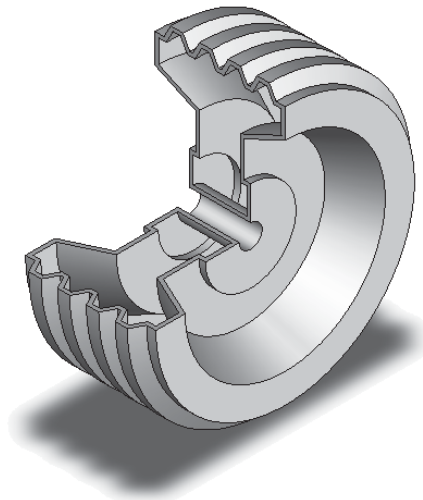
As mentioned earlier, you can assign a material to a model using the **Materials Editor PropertyManager**, which is displayed by choosing the **Edit Material** button. Note that you can also invoke this **PropertyManager** using the **Material** option available in the **Feature Manager Design Tree**.

1. Right-click on the **Material <not specified>** option in the **Feature Manager Design Tree** and choose the **Edit Material** option; the **Materials Editor FeatureManager** is displayed.
2. Click on the + sign located on the left of the **Copper and its Alloys** option from the list box



available in the **Materials** rollout. The tree view expands and the materials available under this family are displayed in the tree view.

3. Select the **Copper** option and choose **OK** from the **Materials Editor PropertyManager**. The model, after assigning the material, is shown in Figure 4-53.



*Figure 4-53 Model after assigning Copper material*

### Saving the Sketch

As the name of the document was specified at the beginning, you just need to choose the save button now to save the document.

1. Choose the **Save** button from the **Standard** toolbar to save the model. If the SolidWorks warning box is displayed, choose **Yes** from it to rebuild the model before saving.

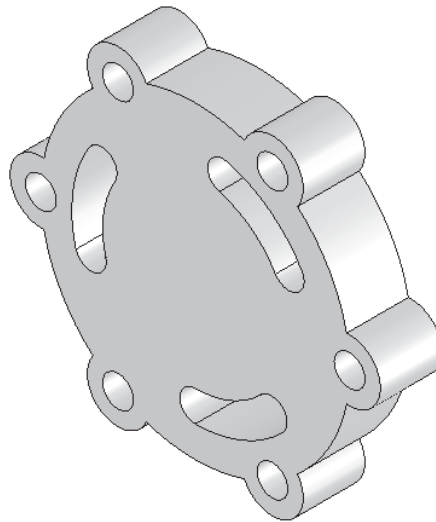
The model is saved with the name `\My Documents\SolidWorks\c04\c04tut2.sldprt`.

2. Choose **File > Close** from the menu bar to close the document.

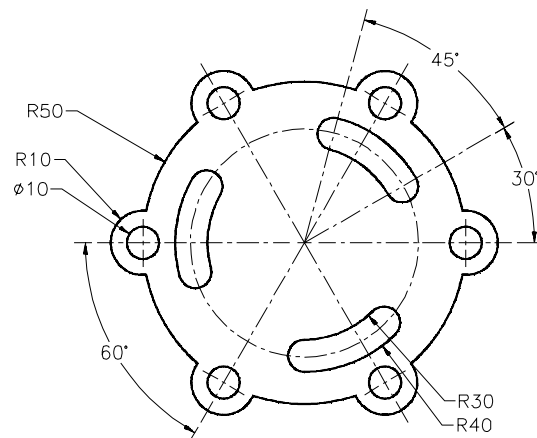
### Tutorial 3

In this tutorial, you will create the model shown in Figure 4-54. Its dimensions are shown in Figure 4-55. The extrusion depth of the model is 20 mm. After creating the model, rotate the view and then change the view back to isometric view before saving the model.

(Expected time: 45 min)



**Figure 4-54** Model for Tutorial 3



**Figure 4-55** Dimensions of the model for Tutorial 3

The steps that will be used to complete this tutorial are given next.

- Start a new SolidWorks part document and then invoke the sketching environment.
- Create the outer loop and then create the sketch of three inner cavities. Finally, draw the six circles inside the outer loop for the holes, refer to Figures 4-56 through 4-60.
- Invoke the **Extruded Boss/Base** tool and extrude the sketch through a distance of 20 mm, refer to Figure 4-61.
- Rotate the view using the **Rotate View** tool.
- Change the current view to isometric view and then save the document.

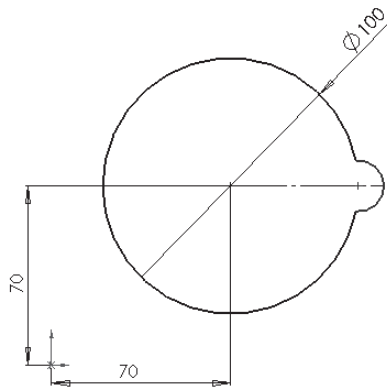
### Starting a New Part Document

1. Choose the **New** button from the **Standard** toolbar and start a new part document using the **New SolidWorks Document** dialog box.
2. Choose the **Sketch** button from the **Standard** toolbar and select the **Front Plane** to invoke the sketching environment.

### Drawing the Outer Loop

When the sketch consists of more than one closed loop, it is recommended that you draw the outer loop first and add relations and dimensions to it so that it is fully defined. Next, draw the inner loops one by one and add relations and dimensions to them. Therefore, you need to first draw the outer loop and add relations and dimensions to it.

1. Draw a circle in the first quadrant and then dimension it so that it is forced to a diameter of 100 mm.
2. Locate the center of the circle at a distance of 70 mm along the X and Y directions from the origin by adding dimensions in both directions. Choose the **Zoom to Fit** button to fit the display on the screen.
3. Draw a horizontal centerline from the center of the circle and then draw a circle with the center point at the intersection of the centerline and the bigger circle.
4. Trim the part of the sketch so that the sketch looks similar to that shown in Figure 4-56.



**Figure 4-56** Sketch after trimming the unwanted portion of circles

5. Dimension the smaller arc so that it is forced to a radius of 10 mm. Add the **Coincident** relation to the center point of the smaller arc and the circumference of the bigger arc.

You will notice that as you add the dimensions and relations to the smaller arc, it turns black. This suggests that the sketch drawn so far is fully defined.

Next, you need to create a circular pattern of the smaller arc. The total number of instances in the pattern is 6 and the total angle is 360-degree.

6. Select the smaller arc and then choose the **Circular Sketch Step and Repeat** button from the **Sketch CommandManager**; the **Circular Sketch Step and Repeat** dialog box is displayed.
7. Drag the center of the circular pattern to the center of the bigger arc.
8. Set the value of the **Number** spinner in the **Step** area to **6**. Accept all other default values and choose the **OK** button to create the pattern.

You will notice that all instances of the pattern are black in color. This is because you have already applied the dimensions and relations to the original instance and so the other instances are also fully defined.

9. Trim the unwanted portion of the bigger arc. This completes the outer loop. The sketch at this stage should look similar to that shown in Figure 4-57.

### Drawing the Sketch of the Inner Cavities

Now, you need to draw the sketch of the inner cavities. Draw the sketch of one of the cavities and then add the required relations and dimensions to it. Next, you need to create a circular pattern of this cavity. The number of instances in the circular pattern is 3.

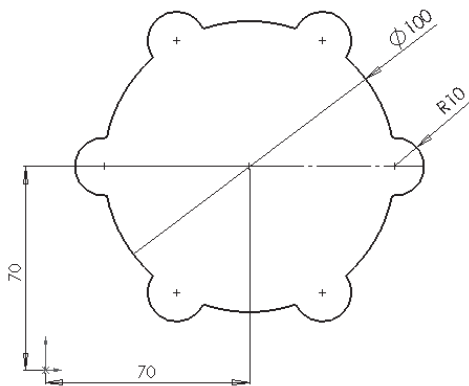
1. Using the **Centerpoint Arc** tool, draw an arc with the center at the center point of the bigger arc of 100 mm diameter.
2. Dimension this arc such that it is forced to a radius of 30 mm. Also, add the angular dimensions to the two endpoints of the arc, refer to Figure 4-58. The arc turns black, suggesting that it is fully defined.
3. Offset the last drawn arc outward through a distance of 10 mm using the **Offset Entities** tool.

The new arc created using the **Offset Entities** tool is also black. Also, a dimension with the value 10 mm is placed between the two arcs.

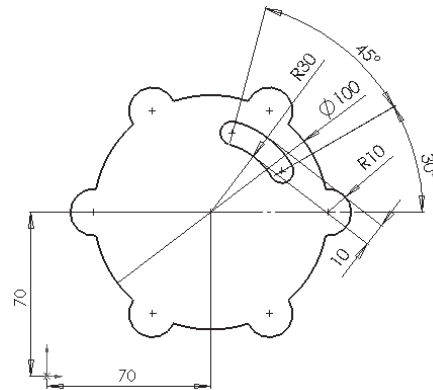
4. Close the two ends of the arc using the **Tangent Arc** tool. This completes the sketch of one of the inner cavities. All entities in the sketch at this stage should be displayed in black, as shown in Figure 4-58.

Next, you need to create a circular pattern of the inner cavity.

5. Select all entities in the sketch of the inner cavity and then invoke the **Circular Sketch Step and Repeat** dialog box.
6. Drag the center of the circular pattern to the center of the outer arc in the sketch.

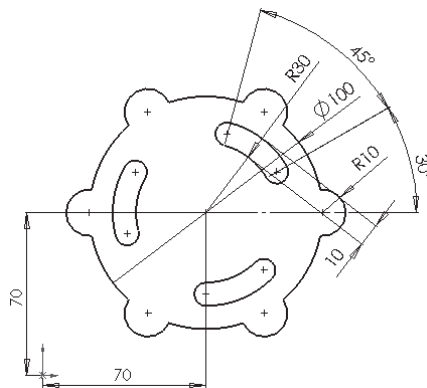


**Figure 4-57** Outer loop of the sketch



**Figure 4-58** Sketch after drawing the sketch of the inner cavity

7. Set the value of the **Number** spinner in the **Step** area to **3** and then choose the **OK** button to create the circular pattern. This completes the sketch of the inner cavities, see Figure 4-59.



**Figure 4-59** Sketch after creating the circular pattern of the inner cavity

### Drawing the Sketch for the Holes

Next, you need to draw the sketch for the holes. Draw one of the circles and then add the dimension to it. Then you need to create a circular pattern of the circle.

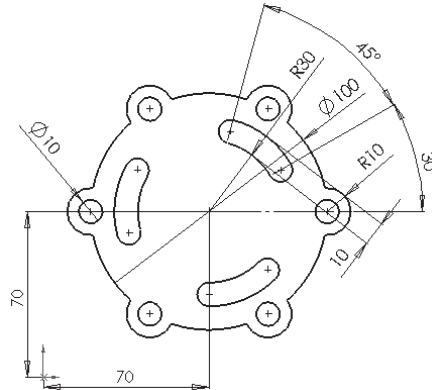
1. Taking the center point of one of the smaller arcs of the outer loop, draw a circle. Add a diameter dimension to it so that the value of the diameter is forced to 10 mm.

The circle turns black when you dimension it.

2. Select the circle and invoke the **Circular Sketch Step and Repeat** dialog box.
3. Drag the center of the circular pattern to the center of the outer arc of 100 mm diameter.

- Set the value of the **Number** spinner in the **Step** area to **6**. Choose **OK** to create the pattern. All instances in the pattern are displayed in black.

This completes the sketch of the model. The final sketch of the model is shown in Figure 4-60.

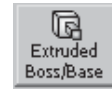


**Figure 4-60** Final sketch of the model

### Extruding the Sketch

The next step after creating the sketch is to extrude it. The sketch is extruded using the **Extruded Boss/Base** tool.

- Choose the down arrow available on the right of the **Features** button in the **CommandManager** to display a flyout. Choose the **Extruded Boss/Base** option from this flyout.



The current view is changed to trimetric view and the **Extrude PropertyManager** is displayed. Also, the preview of the model, as will be created using the default values, is displayed in the drawing area.

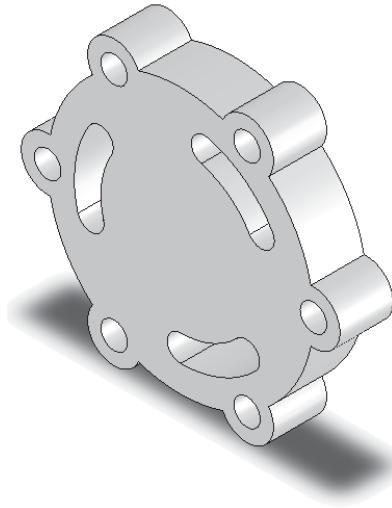
- Set the value of the **Depth** spinner to **20** and then choose the **OK** button to extrude the sketch.

Because the option to display shadow was turned on in the previous tutorial, the shadow is displayed in this tutorial also.

- Press the SPACEBAR and then double-click on the **Isometric** option in the **Orientation** dialog box to change the current view to isometric view. The completed model for Tutorial 3 is shown in Figure 4-61.

### Rotating the View

Before you start rotating the view of the model, it is recommended that you turn off the display of the shadow.



**Figure 4-61** Completed model for Tutorial 3

1. Choose the **Shadows In Shaded Mode** button from the **View** toolbar to turn off the display of the shadow.
2. Now, choose the **Rotate View** button from the **View** toolbar; the arrow cursor is replaced by the rotate view cursor.
3. Press the left mouse button and drag the cursor in the drawing area to rotate the view.
4. Change the current view to isometric view using the **Orientation** dialog box.

### **Saving the Sketch**

1. Choose the **Save** button from the **Standard** toolbar and save the model with the name given below.

`\My Documents\SolidWorks\c04\c04tut3.sldprt.`

2. Choose **File > Close** from the menu bar to close the document.

---

## **SELF-EVALUATION TEST**

Answer the following questions and then compare your answers with those given at the end of this chapter.

1. In SolidWorks, a sketch is revolved using the **Extrude PropertyManager**. (T/F)
2. You can also specify the depth of extrusion dynamically in the preview of the extruded feature. (T/F)

3. You can invoke the drawing display tools such as **Zoom to Fit** while the preview of a model is displayed on the screen. (T/F)
4. When you rotate the view with the current display mode set to **Hidden Lines Removed**, the hidden lines in the model are automatically displayed while the view is being rotated. (T/F)
5. \_\_\_\_\_ tool is used to display the perspective view of a model.
6. The **Cap ends** check box is displayed in the **Extrude PropertyManager** only when the sketch for the thin base feature is \_\_\_\_\_.
7. The \_\_\_\_\_ check box is used to create a feature with different values in both directions of the sketching plane.
8. The \_\_\_\_\_ check box is used to apply automatic fillets while creating a thin feature.
9. The \_\_\_\_\_ button is used to display the shadow in the shaded mode.
10. To resume rotating the view freely after you have completed rotating it around a selected vertex, edge, or face, \_\_\_\_\_ any where in the drawing area.

## REVIEW QUESTIONS

Answer the following questions.

1. You can also invoke the **Rotate View** tool by choosing the **Rotate View** option from the \_\_\_\_\_ that is displayed when you right-click in the drawing area.
2. When you choose the **Wireframe** button, all \_\_\_\_\_ lines will be displayed along with the visible lines in the model.
3. You can also modify the parallel view to perspective view by choosing \_\_\_\_\_ from the menu bar.
4. When you invoke the **Extruded Boss/Base** tool or the **Revolved Boss/Base** tool, the view is automatically changed to a \_\_\_\_\_.
5. The thin revolved features can be created using a \_\_\_\_\_ or an \_\_\_\_\_ sketch.
6. Which of the following buttons is chosen to modify the orientation of the standard views?
  - (a) **Update Standard Views**
  - (b) **Reset Standard Views**
  - (c) None
  - (d) Both



7. Which of the following buttons is not available in the **View** toolbar by default?
- (a) **Hidden Lines Removed**                      (b) **Hidden Lines Visible**  
(c) **Shaded**                                      (d) **Perspective**
8. Which of the following parameters is not displayed in the preview of the model?
- (a) Depth    (b) Draft angle  
(c) None    (d) Both
9. If the sketch is open, it can be converted into
- (a) Thin feature                                      (b) Solid feature  
(c) None    (d) Both
10. In SolidWorks, which tool is used to apply automatic dimensions to the sketch?
- (a) **Autodimension**                              (b) **Smart Dimension**  
(c) None    (d) Both

## EXERCISES

### Exercise 1

Create the model shown in Figure 4-62. The sketch of the model is shown in Figure 4-63. Create the sketch and dimension it using the autodimension option. The extrusion depth of the model is 15 mm. After creating the model, rotate the view. **(Expected time: 30 min)**

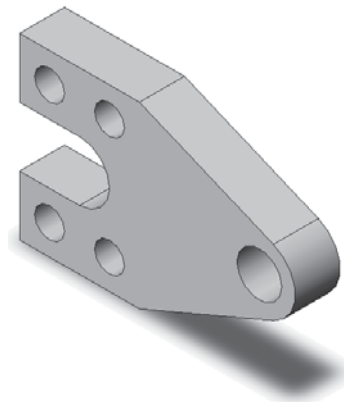


Figure 4-62 Model for Exercise 1

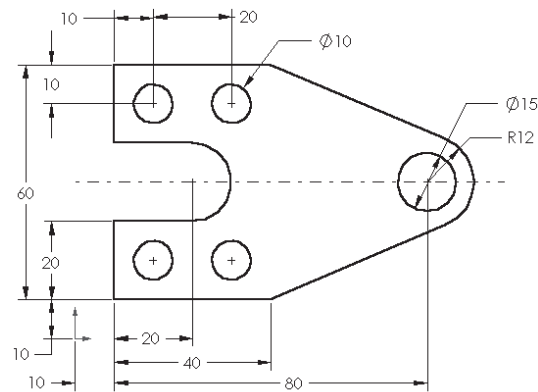
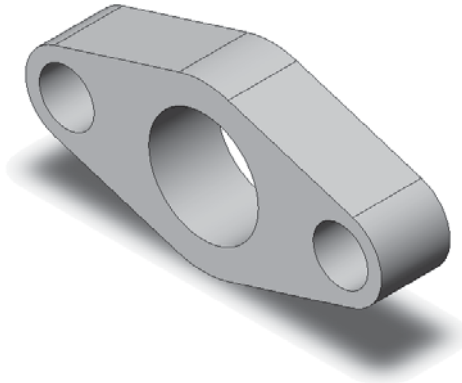


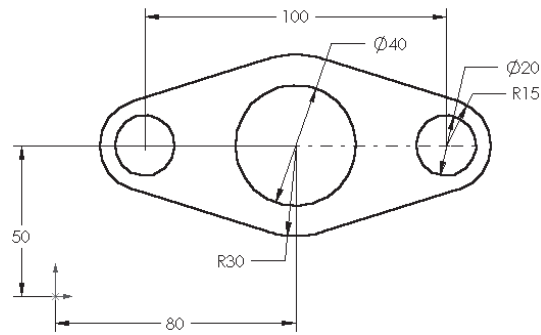
Figure 4-63 Sketch of the model for Exercise 1

## Exercise 2

Create the model shown in Figure 4-64. The sketch of the model is shown in Figure 4-65. Create the sketch and dimension it using the autodimension tool. The extrusion depth of the model is 25 mm. Modify the standard view such that the current front view of the model is displayed when you invoke the top view. **(Expected time: 30 min)**



*Figure 4-64 Model for Exercise 2*



*Figure 4-65 Sketch of the model for Exercise 2*

### Answers to Self-Evaluation Test

1. F, 2. T, 3. T, 4. F, 5. **Perspective**, 6. closed, 7. **Direction 2**, 8. **Auto-fillet corners**, 9. **Shadows In Shaded Mode**, 10. double-click